Simulink^R-SPICE-Interface

Simulink[1] is widely used for the time-domain simulation of control, mechanical, hydraulic and other dynamical systems described by block diagrams. Today, many larger systems contain electrical or electronic subsystems with semiconductor devices. For the simulation of such subsystems, specialized software tools are used. One such program is SPICE[2], a general purpose circuit simulator developed at the University of California at Berkeley which has become a standard in industry, research and education.

The Simulink-SPICE-Interface was developed for the simultaneous simulation of mixed systems containing Simulink-type and SPICE-type subsystems. The interface works within the familiar MATLAB[3]/Simulink environment and is based on a Simulink feature to create new blocks written in C (C MEX S-Functions). The source code of SPICE Version 3F5 was implemented in such a way. The interface block SLSP reads in the circuit file, initializes the simulation, performs the numerical integration in the time-domain and manages the communication between SPICE and Simulink.

To simulate a mixed system, the user first defines the subsystems to be modeled in Simulink (i.e. mechanical, hydraulical, etc.) and the electrical/electronic subsystems to be modeled in SPICE. After selection of the state variables for data exchange between the Simulink- and SPICE-submodels, the Simulink block diagram is edited and the SPICE circuit file is written. After selection of an appropriate integration algorithm and step size, the simulation may be started. Results can be viewed and printed within the MATLAB environment as usual.

The Simulink-SPICE-Interface combines the time-proven tools Simulink and SPICE for a straightforward and efficient simulation of complex mixed systems which contain electrical or electronic subsystems.

Typical applications:

- mechanical/hydraulical/electrical/electronic mixed systems (i.e. automotive)
- aerospace/avionics
- environmental technology
- power system design
- biological/medical/biomedical systems

The Simulink-SPICE-Interface was developed by BAUSCH-GALL GmbH and it is available for MATLAB/Simulink Release 14.

BAUSCH-GALL GmbH, Wohlfartstrasse 21 b,
D-80939 Muenchen, Germany
Telephone: +49/89/3232625, Telefax: +49/89/3231063
email: info@Bausch-Gall.de
web: www.Bausch-Gall.de

d:/slsp/dbl/slspdbl.tex HG 03-NOV-04

[1] The MathWorks, Inc.
[2] Simulation Program with Integrated Circuit Emphasis, U.C.Berkeley, CA, U.S.A.
[3] The MathWorks, Inc.

Example

In this application, the Interface is used for the simulation of the speed control system of a dc motor (Fig. 1). The system block diagram of the overall Simulink-SPICE-Model is shown in Fig. 2 where the shaded areas are modeled in SPICE. The Simulink block diagram is shown in Fig. 3. The time-domain response of the angular velocity $\omega(t)$ is presented in Fig. 6. All other state variables are available for plotting within the MATLAB environment. The operational amplifier in the PI controller is modeled at the transistor level (Fig. 7). The SPICE circuit file was created by using the schematic editor SpiceNet from intusoft.

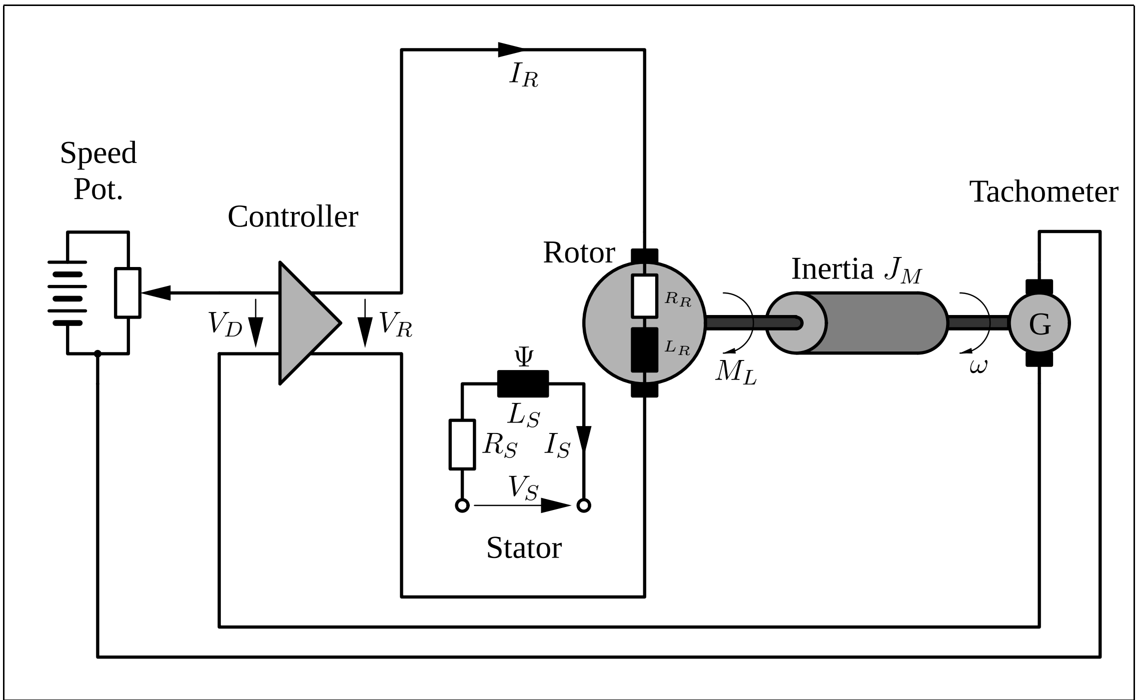

Fig. 1: DC Motor with Control Loop

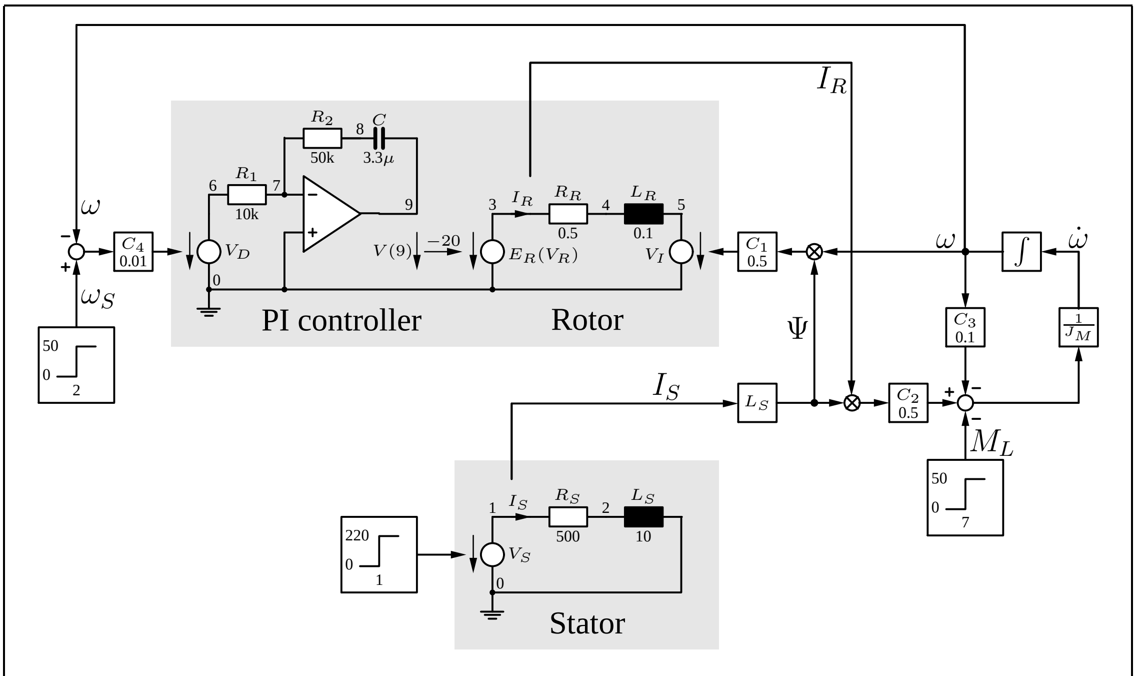

Fig. 2: System Block Diagram of Simulink-SPICE-Model

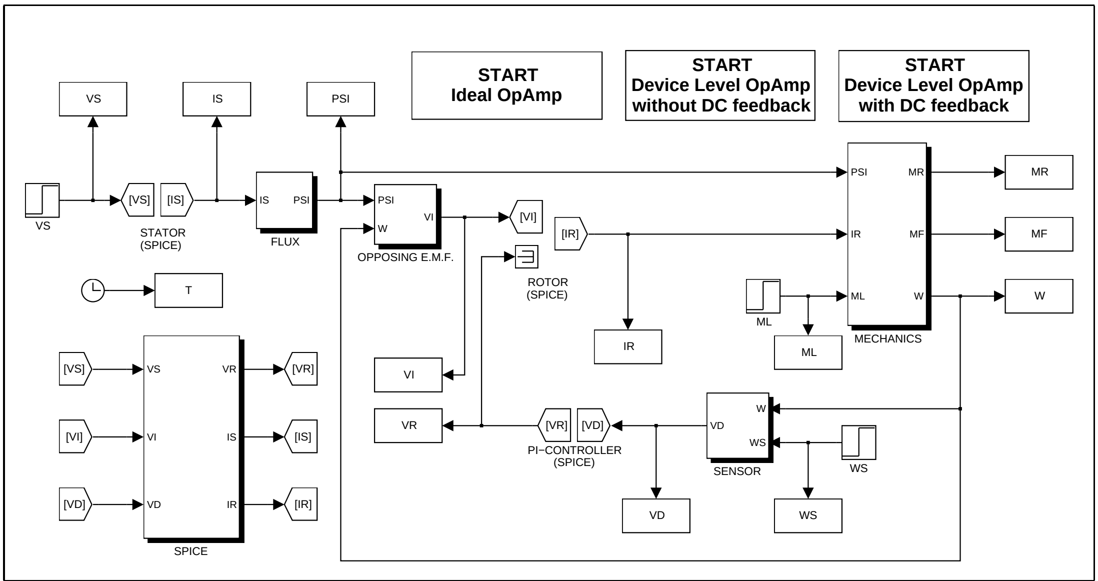

Fig. 3: Simulink block diagram

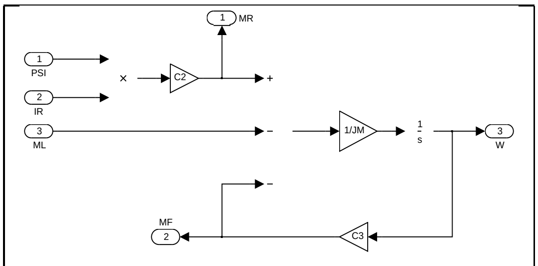

Fig. 4: Subsystem MECHANICS

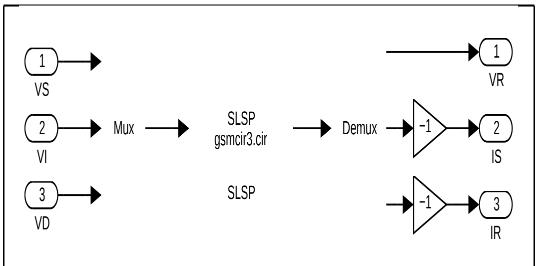

Fig. 5: Subsystem SPICE

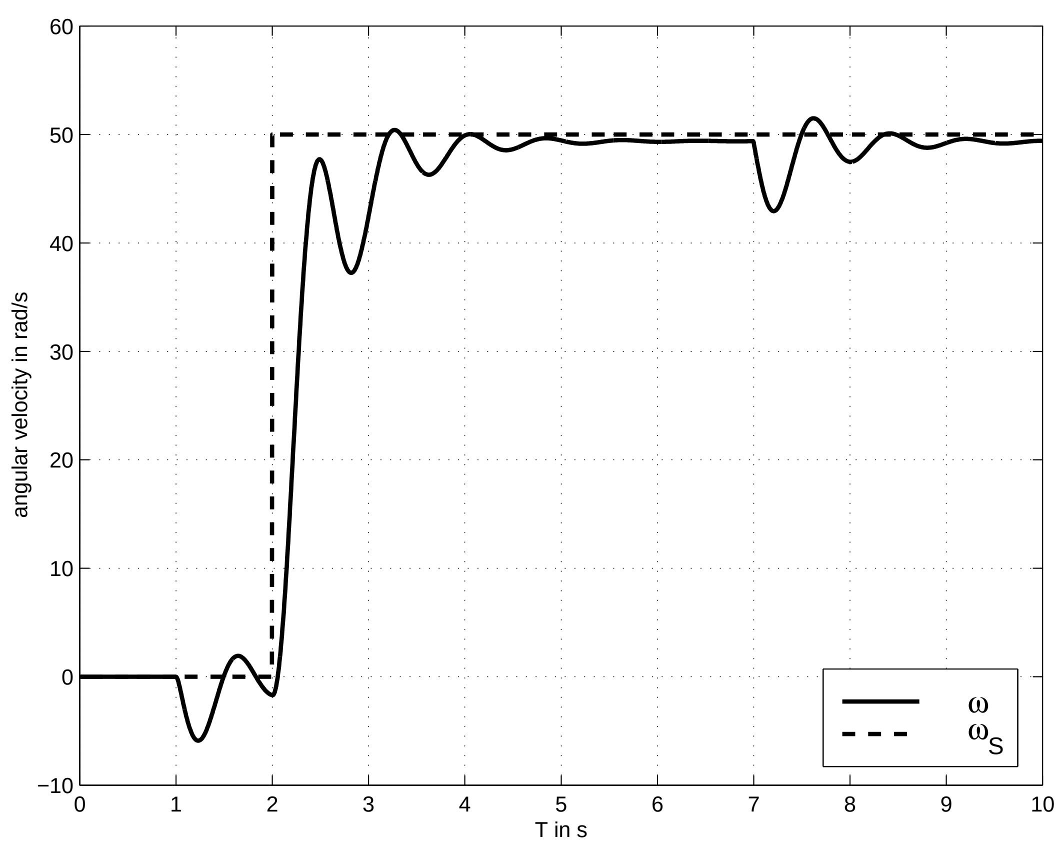

Fig. 6: ω for PI Controller with μA741-Model

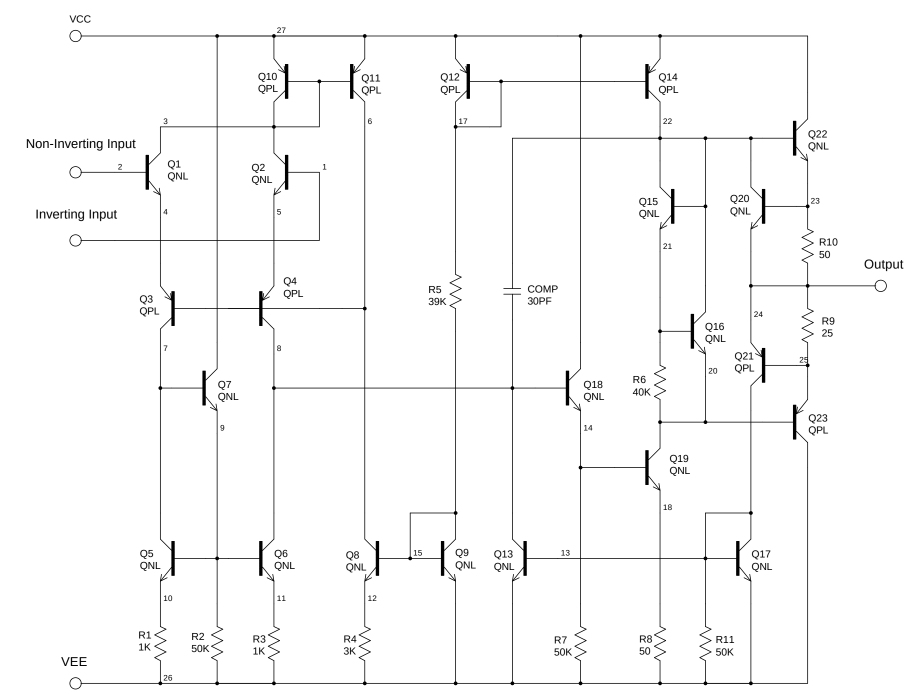

Fig. 7: Device-Level Model of Operational Amplifier μA741

```
File gsmcir3.cir from Chapter 4.5 of SIMULINK-SPICE-Interface User’s Manual
.TRAN 0.005   10    0    0.005
*     TSTEP TSTOP TSTART TMAX (only TMAX important here)
*
* stator
* current IS returned to SIMULINK
* VS set from within SIMULINK
VS 1 0 SLSP
RS 1 2 500
LS 2 0 10
*
* rotor
* current IR returned to SIMULINK
* voltage at node 3 is returned to SIMULINK
ER 3 0 9 0 -20
RR 3 4 0.5
LR 4 5 0.1
* VI set from within SIMULINK
VI 5 0 SLSP
*
* PI controller
* VD set from within SIMULINK
VD 6 0 SLSP
R1 6 7 10K
R2 7 8 50K
R3 12 0 10K
R4 7 9 10MEG
C 8 9 3.3U
VCC 10 0 15
VEE 0 11 15
XOPA 12 7 9 10 11 UA741
*
.SUBCKT UA741 2 1 24 27 26
* subcircuit for device-level model of ua741 opamp
* from [3], pages A1.6 and A1.39
R1 10 26 1K
  .
  .
COMP 22 8 30PF
Q1 3 2 4 QNL
  .
  .
Q23 26 20 25 QPL
.MODEL QNL NPN(BF=80 RB=100 CCS=2PF TF=0.3NS TR=6NS CJE=3PF
+ CJC=2PF VAF=50)
.MODEL QPL PNP(BF=10 RB=20 TF=1NS TR=20NS CJE=6PF CJC=4PF VAF=50)
.ENDS UA741
.END
```

Fig. 8: SPICE circuit file